NUMERICAL SIMULATION OF TRANSIENT FLOW IN STORM SEWERS USING STANDARD AND IMPROVED MCCORMACK SCHEME

MICHAŁ SZYDŁOWSKI
AND JUSTYNA MACHALIŃSKA-MURAWSKA

Faculty of Civil and Environmental Engineering,
Gdansk University of Technology,
Narutowicza 11/12, 80-233 Gdansk, Poland
michal.szydlowski@wilis.pg.gda.pl

(Received 2 June 2012)

Abstract: This paper describes the results of the first part of the research project which aims at developing a hydraulic model for simulation of unsteady flows in storm sewers ranging from gravity flows to surcharged flows resulting with water outflow on the ground surface and propagation of inundation in the flooded area. The paper focuses on the development and assessment of a second-order explicit numerical scheme for unsteady flows in sewers, but only in a single pipe at this moment, without any special elements such as manholes or drop shafts and with no water overflowing problem. The problem of water flow in sewer system pipes is associated with some specific phenomena occurring in conduits during storm events. If the pipes start to be fully filled with water, there is a transition from free surface to pressurized flow. Then, the vice versa effect can be observed. Such transitions are also possible in sewers when the discharge is controlled by control devices, such as gates for example. Moreover, the rapidly varied flow with some hydraulic local effects such as hydraulic jumps or bores can appear during extreme rain episodes. The appropriate modeling techniques have to be applied to solve these problems. The 'Preissmann slot' concept is implemented to simulate the pressurized flow. The original and improved McCormack scheme is used for transcritical flow simulation. The calculated results obtained for some benchmark tests are compared with numerical solutions and laboratory measurements published in the technical literature.

Keywords: mathematical modeling, numerical simulation, transient flow, storm sewers

1. Introduction

The water flow in storm sewers can be usually classified as a free surface open channel flow because water is transported in partially full pipes. However, hydraulic transitions in sewer system pipes can be observed periodically. The transition from the free surface to pressurized flow is a phenomenon occurring in sewers during storms or as a result of the operation of a control device and pumps. Moreover, specific hydraulic effects such as hydraulic jumps or bores can occur in sewer pipes while the water flow is transcritical [1]. The need of simulation of an unsteady transient flow in storm sewers is still a current problem as many operational problems related to rapid pipe filling, water hammer effects, overflowing trough manholes and drop shafts and flooding processes associated with intense rain events are often observed in urban areas. Finally, a storm sewer flow simulation is an element of urban flood modeling and it is necessary for inundation forecasting and risk management in cities. The aim of this work is to test the proposed numerical method and assess its usefulness for modeling water transport in transient flow conditions.

Due to the complexity of the water flow phenomenon in closed conduits, numerical simulation of the flow in a storm sewer system is not a trivial task. Additionally, the storm sewer flow dynamics should be simulated in short time to provide information for storm system operation and flood management. Hence, one dimensional flow models are usually used to describe the unsteady flow in sewer conduits. Many mathematical models of this type of flow, adopting different techniques for implementation of transients can be found in the technical literature. In general, the methods can be grouped into the three main approaches known as rigid column, full dynamic and 'Preissmann slot' models. They have been previously investigated by Wiggert [2], Song *et al.* [3] and Cunge *et al.* [1], respectively. The detailed characteristics of these methods have been presented by Vasconcelos *et al.* [4]. The rigid column methods solve an ordinary differential equation based on the momentum balance in a rigid column represented by the pressurized portion on the flow. The continuity equation is used to obtain the pressurization front location. This method has been implemented by Li and McCroquodale [5]. The idea of similarity between the mathematical description for the open channel and the closed pipe flow is used in the full dynamics models. In this method the pressurization point location is first calculated using the shock fitting technique and then the set of continuity and momentum equations, appropriate for the flow regime, is solved in the pressurized or free surface region of the flow. Such an approach to storm sewer modeling can be found in the works [3, 6, 7]. The third technique known as the 'Preissmann slot' method is very popular in storm sewer modeling, however it can be used only when the flow is expected to be free surface or pressurized, but without subatmospheric pressures. In the Preissmann concept [1] a hypothetical slot at the top of the pipe is introduced to simulate the pressurization effect. The slot makes the flow free surface, even when the pressure line in the pipe cross section is above the top of the pipe. This method is very attractive for numerical modeling as there is no need to track the pressurization front and only one type of equations is solved for all flow conditions. The 'Preissmann slot' method can be found in many hydrodynamic models [8–10]. A similar idea of a conceptual vertical slot may be introduced in all kinds of channels (open or closed) below the conduit bottom for the case of a temporary dry bed. Such a situation can be also observed in storm sewer systems which are nominally dry and convey water occasionally during rain events. The concept of an artificial slot below the channel bottom is called the 'Abbott slot' [11] and allows finding a solution of the Saint Venant equations even if the water depth goes to zero in dry sections of the channel. This technique ensures solution of the flow model despite the fact that the Saint Venant equations are valid only for positive water depth values [12].

In recent times the storm surface flow studies have focused on two additional problems related to the flow pressurization in pipes. The first problem is the prediction of a subatmospheric flow in pressurized flow conditions. Special modeling techniques called the two component pressure approach have been proposed by Vasconcelos *et al.* [4]. A similar solution has been presented by Bourdarias and Gerbi [13]. Some improvements to this approach have been proposed by Sanders and Bradford [14]. The second strongly investigated problem of storm sewer flow modeling is the effect of high pressure transients associated with the interaction between the inflowing water and the air in the pipeline [15].

The solution of water flow equations usually requires that a numerical method is used. If the mathematical model of the flow in storm sewers is composed of continuity and momentum equations the numerous numerical methods for partial differential equations can be used. Numerous methods have been implemented for different models, so far. The method of characteristics (MOC) has been implemented for instance by Li and McCorquodale [5], Song *et al.* [3], Cardle *et al.* [6], Politano *et al.* [7]. The finite difference method (FDM) has been proposed for example by Capart *et al.* [9] who have implemented the Pavia Flux Predictor scheme, or Trajkovic *et al.* [10] using the McCormack scheme. Numerous solutions based on the Finite Volume Method (FVM) have been used in storm sewer modeling, lately. The most popular FVM methods are adopting the Godunov type schemes [16]. Such approaches can be found in the works of Leon *et al.* [15, 16], Bourdarias and Gerbi [13], Vasconcelos *et al.* [4, 17].

In this paper the results of a transient flow simulation in a single pipe using the standard and improved McCormack FDM scheme are presented. The improvement of the scheme is based on the theory of total variation diminishing (TVD) schemes that are capable of capturing sharp discontinuities without generating spurious oscillations. This technique was originally presented by Garcia-Navarro *et al.* [18] and it was proposed for solving one-dimensional open-channel flow equations. The method has been also used to model the dam-break problem by Tseng [19] and to create a hydrodynamic and sediment transport model applicable to steep mountain streams by Papanicolaou *et al.* [20]. In order to assess the quality of the proposed numerical solution, the results of the simulations were compared with the calculations and measurements available in the technical literature.

2. Mathematical model

A numerical simulation of the flow in storm sewers involves the solution of unsteady water flow equations. This set of equations can be derived from the continuity and Navier–Stokes equations. The particularities of storm sewer flows allow some simplifications for the continuity and momentum equations. The most important issue is that storm water is conveyed in a system of channels and tunnels which makes the flow one-dimensional. The conduits are usually prismatic and the system is often composed of circular pipes. Moreover, the inflow into the system is possible only through drop shafts and manholes, hence, there is no lateral inflow between these elements. Therefore, the free surface flow in sewers can be described using the Saint–Venant equations [1] which are generally valid for a gradually varied flow. As during torrential rains various local phenomena such as hydraulic jumps and bores can occur in sewers, a special form of flow model equations should be used. Flow equations in the conservative form should be used in the modeling in order to correctly reproduce the local phenomena with steep water surface fronts [1]. For the case of a prismatic channel such as a pipe with no lateral inflow or outflow the Saint–Venant system, written in the conservative form, can be presented as:

$$\frac{\partial \boldsymbol{U}}{\partial t}+\frac{\partial \boldsymbol{F}}{\partial x}=\boldsymbol{S} \tag{1}$$

where the vectors $\boldsymbol{U}$, $\boldsymbol{F}$ and $\boldsymbol{S}$ are given as:

$$\boldsymbol{U}=\begin{pmatrix} A \\ Q \end{pmatrix}, \quad \boldsymbol{F}=\begin{pmatrix} Q \\ Q^2/A+I \end{pmatrix}, \quad \boldsymbol{S}=\begin{pmatrix} 0 \\ gA(S_0-S_f) \end{pmatrix} \tag{2}$$

where x represents the distance along the sewer conduit, t represents time, A is the cross-sectional wetted area, Q is the flow discharge and g is the gravitational acceleration and $I=pA/\rho$, where p is the fluid pressure at the centroid of A, and ρ is constant fluid density. If the hydrostatic pressure assumption ($p=\rho g h_c$) is valid, the term I can be defined as $I=gAh_c$, where h_c is the distance between the free surface and the centroid of the flow cross-sectional area. The term I is related to the hydrostatic pressure force. S_0 and S_f are the bottom and friction slopes, respectively. The friction slope can be defined by the Manning formula:

$$S_f=\frac{n^2 Q|Q|}{A^2 R_h^{4/3}} \tag{3}$$

where n denotes the Manning friction coefficient, and $R_h=A/P$ is the hydraulic radius and P is the wetted perimeter.

Under the free-surface flow conditions in a circular pipe of inner diameter d, the geometrically related variables can be described by the wetted angle of the pipe θ (Figure 1) as follows [12, 16]:

$$h=\frac{1}{2}[1-\cos(\theta/2)]\,d \tag{4}$$

$$A=\frac{1}{8}[\theta-\sin(\theta)]\,d^2 \tag{5}$$

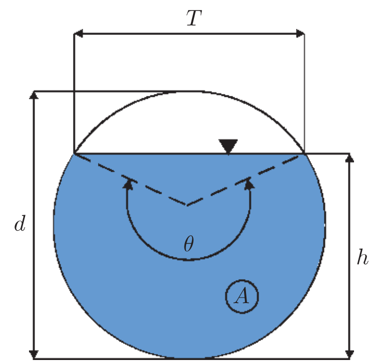

Figure 1. Definition of variables in circular cross section

$$I=\frac{1}{24}\left[3\sin(\theta/2)-\sin^3(\theta/2)-3(\theta/2)\cos(\theta/2)\right]gd^3 \tag{6}$$

$$P=\frac{1}{2}\theta d \tag{7}$$

Moreover, the velocity of the gravity wave defined as:

$$c=\sqrt{\frac{gA}{T}} \tag{8}$$

is also related to the wetted angle θ due to the form A (Equation (5)) and the top width of the free surface T (Figure 1) which can be written as:

$$T=d\sin(\theta/2) \tag{9}$$

The 'Preissmann slot' concept is used in this work to simulate the water flow under pressurized conditions. The idea of a hypothetical slot at the top of the pipe (Figure 2) was presented by Cunge *et al.* [1] and it was implemented for numerical modeling of storm sewers for example by Capart *et al.* [9], Ji [21], Trajkovic *et al.* [10] and Leon *et al.* [17]. The slot assures that after filling the pipe the flow can be treated as an open channel flow despite the fact that the piezometric pressure exceeds the pipe diameter.

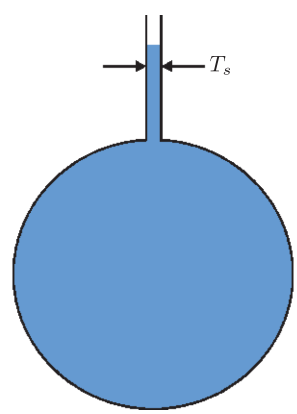

Figure 2. Priessmann slot

In such a situation the velocity of the gravity wave in an open channel with a slot can be expressed according to Equation (8) as:

$$c = \sqrt{\frac{gA_p}{T_s}} \tag{10}$$

where A_p is the cross-sectional area of the pipe and T_s is the width of the slot. Moreover, the velocity c for the pressurized flow must be the same as the sound velocity a in a closed pipe. This assumption can be used to determine the appropriate width of the slot. The choice of the slot size affects the stability of numerical computations, thus it is an individual problem for each simulation, nonetheless, finally the 'Preissmann slot' idea makes the transient flow simulation possible without the need to separately track the pressurization front. Nevertheless, the solution of water flow equations based on the Priessmann concept of a hypothetical slot has few main disadvantages as has been reported by Vasconcelos *et al.* [4]. The most important issue is the inability to simulate a full pipe subatmospheric flow, like in some special segments of sewers of the siphon type. In such places the piezometric head drops below the pipe crown and the 'Preissmann slot' idea results in the free surface flow in the pipe, what is not true. Other problems are the mass conservation due to an additional slot in the pipe and the simulation instability, when water exceeds the pipe diameter and enters the slot.

3. Numerical solution

The Saint–Venant Equations (1)–(2) are a system of partial differential equations and their solution for the given initial and boundary conditions is composed of the functions $A(x,t)$ and $Q(x,t)$. A numerical method must be applied to solve the Saint–Venant equations for complex sewer geometry and hydraulic conditions. In this paper the finite differences method (FDM) has been chosen to integrate the model equations in space and time [1]. The FDM schemes discretize continuous space and time into a grid system, and the variables are evaluated at separate nodes of the grid. For simple FDM schemes, the first-order derivatives are approximated with either central, backward, or forward differences, while the second-order derivatives are approximated with central differences. After discretization of integration space, the selected time level can be represented as the time $t_n = n \cdot \Delta t$, and each point in space (along the channel length) defines the computing node $x_i = (i-1)\cdot\Delta x$, where Δt is the time increment and Δx is the uniform mesh size.

There are numerous FDM numerical schemes which can be used for the solution of the Saint–Venant equations. The McCormack scheme can be used to ensure the second order accuracy of derivatives approximation in space and time and keep the simplicity of the calculation [22]. In this study, the original and improved McCormack schemes are investigated. The improved scheme is based on the technique originally proposed by Garcia-Navarro *et al.* [18] for open channel flow simulation. The main advantage of the original scheme is the ability to calculate the gradually and rapidly varied flow, what is needed to simulate the water flow in sewer conduits. Moreover, the inclusion of the source terms is relatively simple and it is suitable for implementation in an explicit time-marching algorithm. On the contrary, spurious oscillations usually appear in numerical solutions using the McCormack scheme. An improvement based on the TVD theory is proposed in this paper to avoid or reduce this problem for transient flow in storm sewer modeling. The concept of TVD schemes has been introduced by Harten and Hyman [23]. For certain types of equations TVD algorithms ensure that the total variation (TV) does not increase with time, *i.e.*:

$$TV\left(\boldsymbol{U}^{n+1}\right) \leq TV\left(\boldsymbol{U}^{n}\right) \quad \Rightarrow \quad \sum_{i} |\boldsymbol{U}_{i+1}^{n+1} - \boldsymbol{U}_{i}^{n+1}| \leq \sum_{i} |\boldsymbol{U}_{i+1}^{n} - \boldsymbol{U}_{i}^{n}| \tag{11}$$

Such an improved McCormack scheme is a shock-capturing technique with a second-order accuracy both in time and space in non-critical sections, but it switches the accuracy to the first-order at extreme points.

The algorithm based on the McCormack original scheme involves a two-step procedure known as the predictor-corrector method and it can be presented as:

$$\boldsymbol{U}_{i}^{p} = \boldsymbol{U}_{i}^{n} - \frac{\Delta t}{\Delta x}\left(\boldsymbol{F}_{i+1}^{n} - \boldsymbol{F}_{i}^{n}\right) + \Delta t \boldsymbol{S}_{i}^{n} \tag{12}$$

$$\boldsymbol{U}_{i}^{c} = \boldsymbol{U}_{i}^{n} - \frac{\Delta t}{\Delta x}\left(\boldsymbol{F}_{i}^{p} - \boldsymbol{F}_{i-1}^{p}\right) + \Delta t \boldsymbol{S}_{i}^{p} \tag{13}$$

where the superscript $p(c)$ refers to the predictor (corrector) step and n is the time level.

The final updating formula, representing the solution at the next time level $(n+1)$ has the form:

$$\boldsymbol{U}_{i}^{n+1} = \frac{1}{2}\left(\boldsymbol{U}_{i}^{p} + \boldsymbol{U}_{i}^{c}\right) \tag{14}$$

The McCormack scheme is an explicit technique of the FDM, therefore, it has to satisfy the Courant–Friedrich–Lewy (CFL) criterion at each grid point i in order to be stable [22]. The CLF criterion is defined as:

$$\mathrm{Cr} = \frac{|Q/A| + c}{\Delta x / \Delta t} \leq 1 \tag{15}$$

where Cr is the Courant number at point i.

The TVD McCormack scheme is an extension of the original method and it includes a shock-capturing technique capable of rendering the solution oscillation. The improved scheme involves an additional computational term in the updating step of the original predictor corrector procedure (14) [18], which can be written following Tseng [24] as:

$$\boldsymbol{U}_{i}^{n+1} = \frac{1}{2}\left(\boldsymbol{U}_{i}^{p} + \boldsymbol{U}_{i}^{c}\right) + \frac{1}{2}\frac{\Delta t}{\Delta x}\left(\boldsymbol{R}_{i+1/2}\boldsymbol{\Phi}_{i+1/2} - \boldsymbol{R}_{i-1/2}\boldsymbol{\Phi}_{i-1/2}\right) \tag{16}$$

The second term in Equation (16) which has to be calculated at intermediate states between the grid points $i-1$, i and $i+1$, what will be described later, generally equips the scheme with TVD properties adding a numerical dissipation to the original scheme. Due to this modification, the scheme retains a second-order accuracy in space and time for continuous regions [25] and it is able to limit the solution oscillations near the extremes by reducing the accuracy to the first-order in these sections.

The TVD improvement requires the quasi linear form of the Saint–Venant equations (1)–(2) to calculate the additional term in formula (16). The original problem (1) can be transformed to the following form:

$$\frac{\partial \boldsymbol{U}}{\partial t}+\boldsymbol{A}\frac{\partial \boldsymbol{U}}{\partial x}=\boldsymbol{S} \tag{17}$$

where $\boldsymbol{U}$ is the same as in Equation (1) and the Jacobian matrix $\boldsymbol{A}$ of $\boldsymbol{F}$ with respect to $\boldsymbol{U}$ can be written as:

$$\boldsymbol{A}=\begin{bmatrix} 0 & 1 \\ c^2-\frac{Q^2}{A^2} & 2\frac{Q}{A} \end{bmatrix} \tag{18}$$

where c is the velocity of the wave (celerity) defined by Equation (8). The Jacobian matrix $\boldsymbol{A}$ is diagonalizable, hence, the following equation has to be satisfied:

$$\boldsymbol{A}=\boldsymbol{R}\,\boldsymbol{\Lambda}\,\boldsymbol{L} \tag{19}$$

where $\boldsymbol{\Lambda}$ is a diagonal matrix containing the eigenvalues of matrix $\boldsymbol{A}$, whereas $\boldsymbol{R}$ and $\boldsymbol{L}$ contain associated right and left eigenvectors. The eigenvalues λ of matrix $\boldsymbol{A}$ can be evaluated by the solution of the characteristic equation [26]:

$$|\boldsymbol{A}-\lambda \boldsymbol{I}|=0 \tag{20}$$

where $\boldsymbol{I}$ is the identity matrix. Considering the Jacobian matrix (18) the roots of (20) equal:

$$\lambda_1=u-c \tag{21a}$$

$$\lambda_2=u+c \tag{21b}$$

where $u=Q/A$. The matrix $\boldsymbol{\Lambda}$ and the corresponding right, used in updating the step of the TVD McCormack scheme (16), and left eigenvector matrices for matrix $\boldsymbol{A}$ are defined as:

$$\boldsymbol{\Lambda}=\begin{bmatrix} \lambda_1 & 0 \\ 0 & \lambda_2 \end{bmatrix} \tag{22a}$$

$$\boldsymbol{R}=\begin{bmatrix} 1 & 1 \\ \lambda_1 & \lambda_2 \end{bmatrix} \tag{22b}$$

$$\boldsymbol{L}=\frac{1}{2c}\begin{bmatrix} -\lambda_2 & 1 \\ \lambda_1 & -1 \end{bmatrix} \tag{22c}$$

The two components of vector $\boldsymbol{\Phi}_{i+1}$ in Equation (16), which are evaluated at the intermediate state between the grid points i and $i+1$, are defined as:

$$\Phi^k_{i+1/2}=\psi\left(\lambda^k_{i+1/2}\right)\left(1-\frac{\Delta t}{\Delta x}\left|\lambda^k_{i+1/2}\right|\right)\left(1-\varphi\left(r^k_{i+1/2}\right)\right)\alpha^k_{i+1/2}, \quad k=1,2 \tag{23}$$

The function Ψ is an entropy correction to the eigenvalues preventing the appearance of unphysical flow discontinuities, those in which energy increases across the shock. In the simplest form it can be written as [18]:

$$\Psi(\lambda)=\begin{cases} |\lambda| & \text{if } |\lambda|\geq\varepsilon \\ \varepsilon & \text{if } |\lambda|<\varepsilon \end{cases} \tag{24}$$

where ε is a small positive number (from 0.1 to 0.3 [18]) the value of which has to be determined for each individual problem. Formulas for the evaluation of ε and other forms of entropy correction have been proposed by Harten and Hyman [23].

The characteristic variable α in formula (23) is defined as:

$$\alpha_{j+1/2} = \frac{1}{2c_{i+1/2}} \begin{bmatrix} -\lambda_2 & 1 \\ \lambda_1 & -1 \end{bmatrix}_{i+1/2} \begin{bmatrix} A_{i+1} - A_i \\ Q_{i+1} - Q_i \end{bmatrix} \tag{25}$$

The averaging procedure proposed by Roe [27] can be applied to calculate the mean values of the flow parameters in (25) which have to be determined at the intermediate point $i+1/2$. Following Garcia-Navarro *et al.* [18], the discrete approximations to the local water velocity and wave celerity can be presented as:

$$u_{i+1/2} = \frac{Q_{i+1}/\sqrt{A_{i+1}} + Q_i/\sqrt{A_i}}{\sqrt{A_{i+1}} + \sqrt{A_i}} \tag{26a}$$

$$c_{i+1/2} = \frac{c_i + c_{i+1}}{2} \tag{26b}$$

The limiter parameter has to be incorporated into the solution procedure to obtain non-oscillatory solutions in regions where some flow discontinuities like hydraulic jumps or bores occur. In Equation (23) the function φ is a limiter parameter and it is responsible for adding artificial dissipation to the numerical solution in regions of steep gradients. The numerical dissipation makes the solution monotone at extreme points. Little or no dissipation is added in the continuous regions of smooth variation. Many forms of the limiting function can be found in the literature. Their review in relation to the water flow problem has been presented by Toro [16]. Following Tseng [24] the minmod limiter is used in this paper to simulate transient flow in storm sewers. This function can be written as:

$$\varphi\left(r^k_{i+1/2}\right) = \begin{cases} \min\left(\left|r^k_{i+1/2}\right|, 1\right) & \text{if } r^k_{i+1/2} > 0 \\ 0 & \text{if } r^k_{i+1/2} \leq 0 \end{cases} \tag{27}$$

where r is the ratio of the characteristic variables estimated as follows:

$$r^k_{i+1/2} = \frac{\alpha^k_{i+1/2-s}}{\alpha^k_{i+1/2}}, \quad s = \operatorname{sign}\left(\alpha^k_{i+1/2}\right) \tag{28}$$

It is necessary to specify additional conditions to perform numerical simulation of the Saint–Venant Equations (1)–(2). According to the theory of solving partial differential equations the conditions include the initial condition and boundary conditions [1]. The initial water surface profile which determines the cross sectional area A and the flow rate Q at time $t=0$ has to be known along the channel before starting simulation of an unsteady flow in sewer conduits. The type and number of boundary conditions result from the characteristics theory [1] what is associated with the variability of flow parameters at inflow and outflow cross-sections of the channel. In accordance with the characteristics theory the number of conditions imposed at the boundary depends on the local value of the Froude number and the flow direction. Analyzing the schematic (Figure 3) of the characteristic curves of Equations (1)–(2) it can be seen that for subcritical flow ($\mathrm{Fr} < 1$) it is always one characteristic that enters the calculation domain through the inflow or outflow section of the channel, hence, it is only one condition that is required at each boundary. For supercritical inflow ($\mathrm{Fr} > 1$) both characteristics enter the calculation domain through the inflow section, hence, two conditions have to be imposed there, but no condition is needed at the outflow channel section.

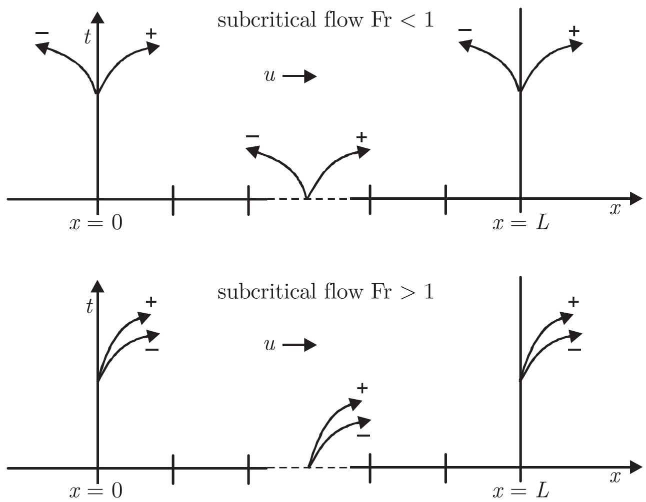

Figure 3. Schematic of characteristic curves for Saint–Venant equations

The direction and regime of the water flow in storm sewers are variable what determines the number and type of the boundary conditions needed at the inlet or outlet sections of conduits. There are many types of junctions in storm sewers. Usually sewer pipes are connected with each other inside manholes which function as small reservoirs determining the flow between the manhole and the pipe. Numerous situations can be observed at this point what is in relation to the piezometric head levels in pipes and manholes. The proposition of the organization of possible boundary flow regimes for storm sewer modeling has been proposed by Capart *et al.* [9]. It has been extended by Sanders and Bradford [14], lately. A detailed analysis of the boundary conditions is not substantial for this study, therefore it has been disregarded.

4. Numerical simulations

In this section, the numerical results obtained using the original and improved McCormack scheme for four test problems are presented and analyzed. All the examples consider the flow transients in circular pipes resulting from sudden maneuvers of flow control devices during laboratory experiments. These four test cases can be found in the literature and they are often used as benchmark tests for the analysis of numerical solutions. However, as no exact laboratory data are available for the authors of this paper, the comparison between the authors' own calculations and the published measurements is not a true verification procedure, but can be seen as a representative test of the numerical simulation of sewer pressurization.

The first test was analyzed and described by Capart *et al.* [9]. It consisted in the pressurization of a circular pipe on a steep slope. The experimental set-up featured a pipe 12.74m long and 0.145m in inner diameter. The Manning roughness coefficient was $0.009 \mathrm{s/m}^{1/3}$. The pipe consisted of three segments with different bottom slopes of 0.01954 (0–3.48m), 0.01704 (3.48–9.23m) and 0.01255 (9.23–12.74m), respectively and it connected two tanks. The water level in the downstream tank was kept below the pipe outlet. The upstream tank supplied the pipe with water. The experiment started from a steady state. A constant discharge of $0.0042 \mathrm{m}^3/\mathrm{s}$ was kept at the upstream. Due to the relatively steep slope of the pipe, the flow at the inlet was supercritical while the flow regime at the downstream end depended on the water level which varied during the experiment. The hydraulic jump was generated by a sudden gate closure at the downstream end of the pipe in the first phase of the experiment. Then, the jump was going upstream resulting in pressurization of the pipe. When the jump was near the upstream end, the downstream gate was open, leading to a sudden decrease in the water surface. In this second phase a fast transient, in the form of a negative wave, returned the flow to its initial condition.

For the numerical simulation of Test No.1 the pipe was discretized into 255 nodes with the spatial step length $\Delta x = 0.05 \mathrm{m}$. The simulation was carried out with the time step of $\Delta t = 0.005 \mathrm{s}$, ensuring the stability of the solution. The varying bottom line of the pipe was replaced by a constant average value of the bottom slope (0.017) in the simulation. The comparison between the results obtained by Capart *et al.* [9] and the authors' own calculations is presented in Figures 4–9. The graph in Figure 4 shows the variation in time of the piezometric head for three measuring points located 3.06m (C3), 5.50m (C4) and 7.64m (C6) from the upstream end of the pipe, which were originally presented by Capart *et al.* [9]. The computed results presented in this picture correspond to the solution obtained using the Pavia Flux Predictor scheme. The authors' own numerical calculations for the same control points are presented in Figure 5. It can be observed that the water levels and the velocity of the wave front seem to be in good agreement. However, it can be seen that the results obtained with the classic and improved McCormack schemes differ from each other. The standard scheme produces spurious oscillations near sharp pressure (water level) fronts, while the improved version of the scheme ensures quite a smooth solution.

A comparison between the piezometric profiles computed and observed during the first period of the experiment is presented in Figures 6–7. The graphs present the piezometric lines at the same times. The hydraulic jump formation and progression can be seen in these pictures. Analyzing the piezometric profiles for different time steps it can be seen that the free surface water flow exists along the pipe at the first moment after the jump. Then, the pressurization process

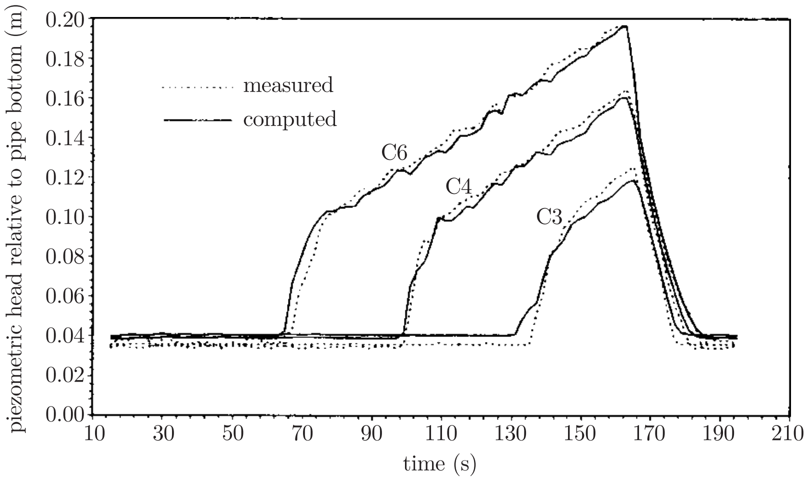

Figure 4. Test No. 1: Piezometric head measured and calculated at sections C3, C4 and C6 by Capart *et al.* [9]

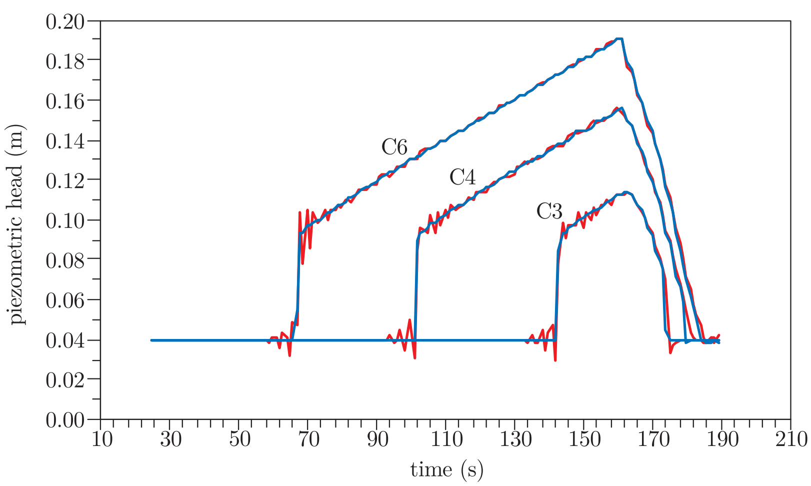

Figure 5. Test No. 1: Piezometric head calculated at sections C3, C4 and C6 using classic (red line) and improved (blue line) McCormack scheme

can be observed with the rising water level at the outflow section of the pipe. At the same time the hydraulic jump migrates upstream. In the next two figures (Figures 8–9) the abrupt transient resulting from sudden opening of the gate and water release can be observed. This process is very fast and it makes the flow rapidly varied. It can be seen that the agreement between the originally measured and computed results and the results obtained using the McCormack schemes is quite good, at least at the qualitative level. However, some differences between the shapes of the water surface profile in the pipe can be observed when the free surface flow occurs. The reason for this discrepancy cannot be precisely explained at this moment due to lack of detailed information about the boundary conditions at the outlet section of the pipe after gate opening. Finally, it can be found again that the solution obtained using the improved McCormack scheme better fits the previously published results than its classical version.

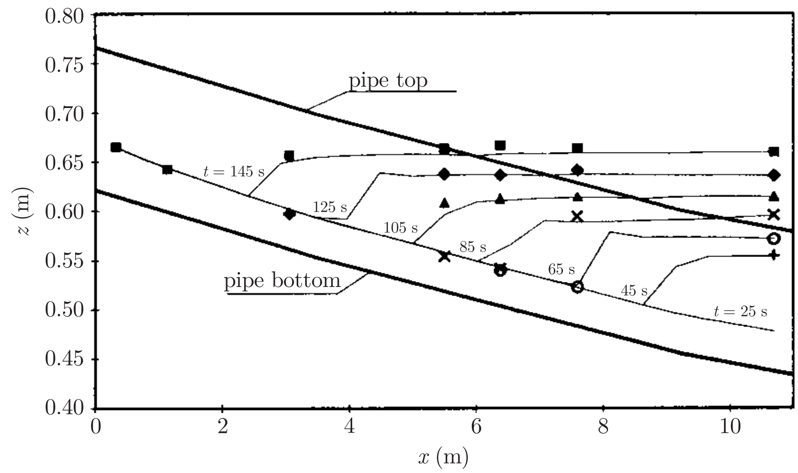

Figure 6. Test No. 1: Piezometric profiles measured (markers) and calculated (solid line) by Capart *et al.* [9] for first period of experiment

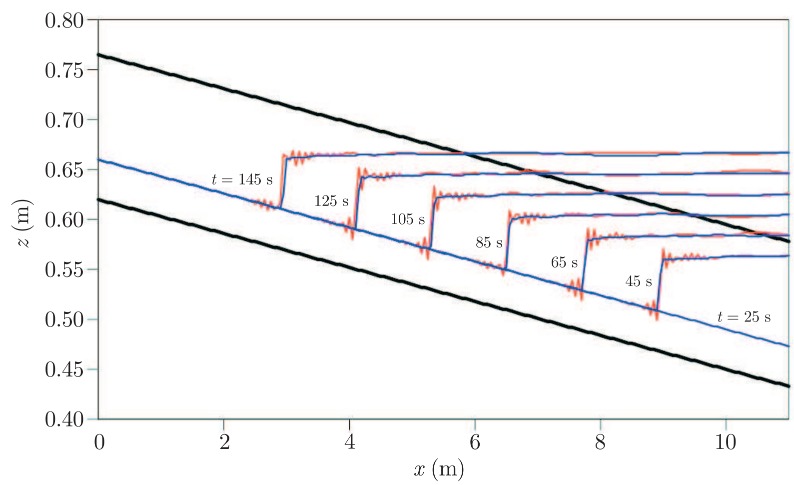

Figure 7. Test No. 1: Piezometric profiles calculated using classic (red line) and improved (blue line) McCormack scheme for first period of experiment

The second and third test cases, examined in this paper, are described by Politano *et al.* [7]. The former presents the laboratory experiments carried by Cardle *et al.* [6] and the latter concerns the measurements obtained by Trajkovic *et al.* [10]. The numerical results presented by Politano *et al.* [7] were calculated using the Interface Tracking Method (ITM).

The experiment of Cardle *et al.* [6] (Test No. 2) was performed in a circular pipe 48.8m long, 0.1626m inner diameter and with a slope of 0.001. The Manning

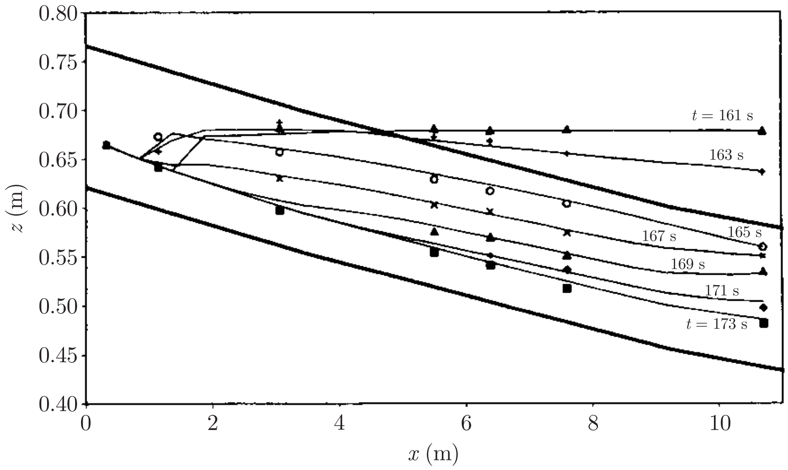

Figure 8. Test No. 1: Piezometric profiles measured (markers) and calculated (solid line) by Capart *et al.* [9] for second period of experiment

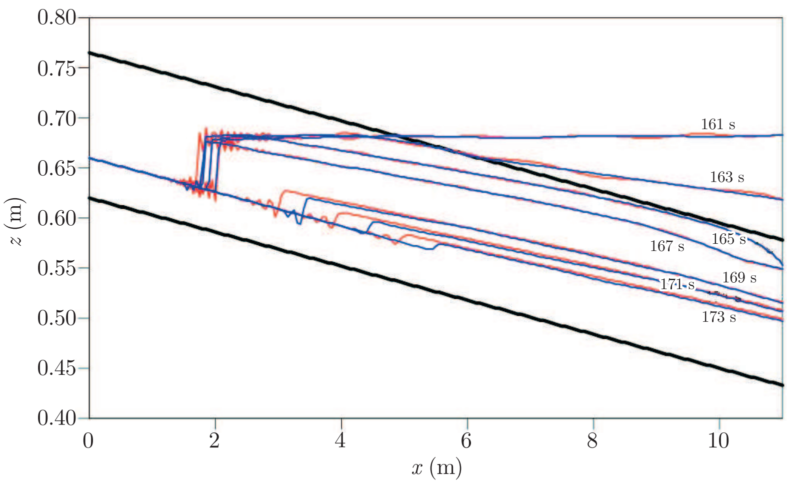

Figure 9. Test No. 1: Piezometric profiles calculated using classic (red line) and improved (blue line) McCormack scheme for second period of experiment

roughness coefficient was identified as equal to $0.012\mathrm{s/m}^{1/3}$. The pipe flow pressurization and the water surface front migrating upstream the pipe were generated by a sudden closure of the gate at the outflow section. The variation in the piezometric head in time was measured at three points along the pipe located at 9.1 m (P1), 21.3 m (P2) and 39.6 m (P3) from the downstream end of the pipe. The experiment started from the steady state. At the upstream section, the constant inflow rate was equal to $0.0068\mathrm{m}^3/\mathrm{s}$. The water level in the downstream reservoir was initially at an elevation of 0.15 m.

For the numerical flow simulation of Test No. 2 the pipe was discretized into 245 nodes with the spatial step length of $\Delta x = 0.2$ m. The simulation was carried out with the time step equal to $\Delta t = 0.01$s, ensuring the stability of the solution. The comparison between the measurements and the numerical results presented by Politano *et al.* [7] and the authors' own simulation are presented in Figures 10–11, respectively. It can be seen that the piezometric head measured at the control points P1, P2 and P3 and the velocity of the wave front are in good agreement with the calculations obtained using the improved McCormack scheme. The standard McCormack scheme produces oscillations near the piezometric head front which were not observed during the experiment. These spurious oscillations make the solution unphysical and the results are not acceptable. Moreover, the observed increase in the piezometric head is not as abrupt as the simulated one. This might be a result of the finite time of the gate closure in the laboratory experiment while the simulation assumes a sudden shutoff of the flow.

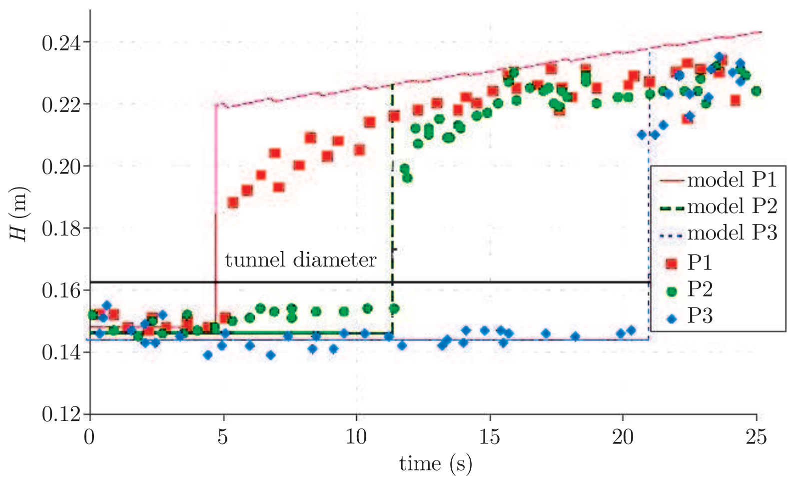

Figure 10. Test No. 2: Piezometric head measured by Cardle *et al.* [6] (markers) and calculated by Politano *et al.* [7] (lines)

Additionally, the evolution of the piezometric profile in time is shown in Figure 12. It can be observed that the results obtained using the improved McCormack scheme are better than the standard solution. Again, the spurious oscillations, observed near the pressure front advancing upstream the pipe, make the solution unsatisfactory.

The next test case (Test No. 3) concerns the laboratory experiment carried out by Trajkovic *et al.* [10]. Figure 13 shows the comparison between the measurements and the piezometric heads computed by Politano *et al.* [7]. The experimental analysis of the transient flow was performed in a circular pipe 10 m long, 0.10 m in inner diameter and a slope of 0.027. The Manning roughness coefficient was equal to $0.008\,\mathrm{s/m^{1/3}}$. The pressurization of the pipe flow was generated by closing the sluice gate suddenly at the downstream end. The variation of the piezometric head in time was measured at two points along the pipe located at 0.6 m (P1) and 4.5 m (P2) from the downstream end of the

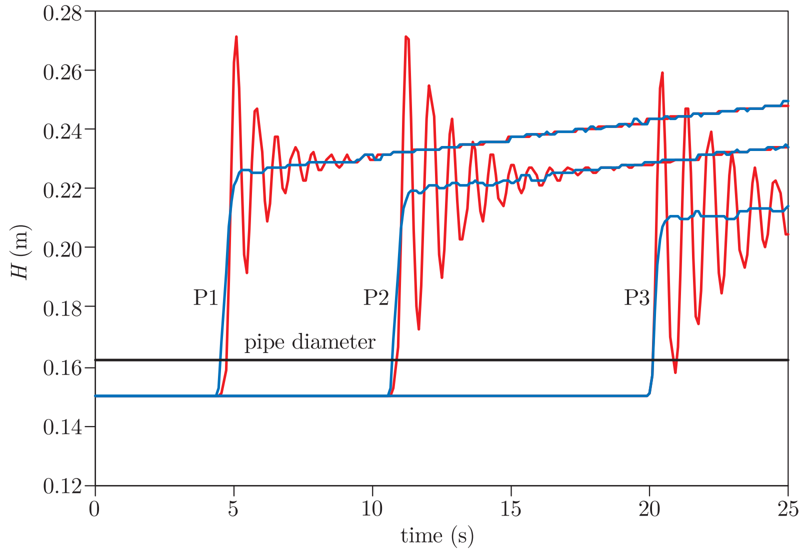

Figure 11. Test No. 2: Piezometric head calculated using classic (red line) and improved (blue line) McCormack scheme

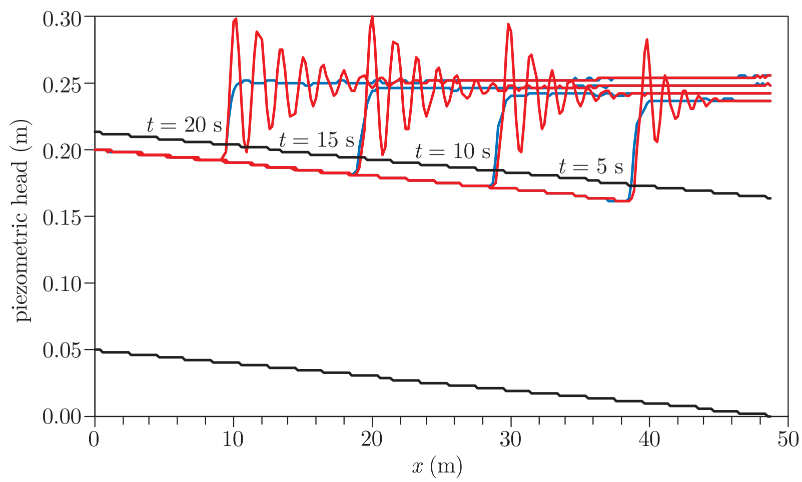

Figure 12. Test No. 2: Piezometric profiles calculated using classic (red line) and improved (blue line) McCormack scheme

pipe. The experiment started from a steady supercritical free surface flow with a discharge of $0.0013\,\mathrm{m}^3/\mathrm{s}$.

The pipe was discretized into 201 nodes with the spatial step length of $\Delta x = 0.05\,\mathrm{m}$ in order to simulate the transient flow. The simulation was carried out with the time step of $\Delta t = 0.001\,\mathrm{s}$, ensuring the stability of the solution. Figures 14–15 present the results of the authors' own simulation. It can be observed that the piezometric head measured at control points P1 and P2 and the velocity of the wave front are in good agreement with the calculations obtained using the improved McCormack scheme. The main features of the transient flow are predicted by the model quite well. In this test case, similarly as in Test No. 1, the hydraulic jump, migrating upstream the pipe, does not pressurize the pipe. The pressurization takes place from a subcritical free surface flow condition. Moreover, as in the previous simulations, the calculated front of the piezometric head is steeper than the observed one due to the assumption of a sudden shutoff of the flow at the outflow section. Again, the solution obtained using the standard McCormack scheme is distorted by oscillations that make the results useless for transient flow analysis.

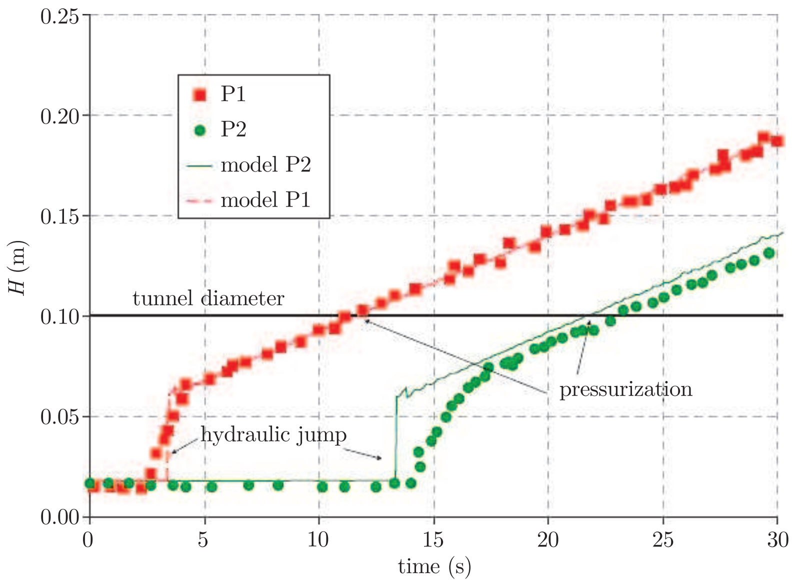

Figure 13. Test No. 3: Piezometric head measured by Trajkovic *et al.* [10] (markers) and calculated by Politano *et al.* [7] (lines)

The last analysis (Test No. 4) is related to the laboratory experiment described by Fuamba [28]. In this experiment the laboratory set-up was composed of a pipe 6m long, which was supplied from the upper tank. The pipe was 0.1m in inner diameter and the slope was 0.0013. A control valve to maintain the flow was located at the inflow section. A sliding weir was operated downstream to control the downstream water depth in such a way that the weir height was 0.085m. The experiment analyzed in this test case differs from previous examples. In this test, the pipe pressurization is not an effect of the hydraulic jump generated at the outflow section but it is a result of an increase in the flow rate controlled at the inflow section. The experiment was carried out as follows [28]. Starting from a complete rest (initial condition), the valve at the inflow section was gradually opened. Then, the water depth at the upstream end increased until the pipe invert was reached. The pipe became initially submerged at the upstream cross-section. A surge front formed then and started propagating downstream, forming

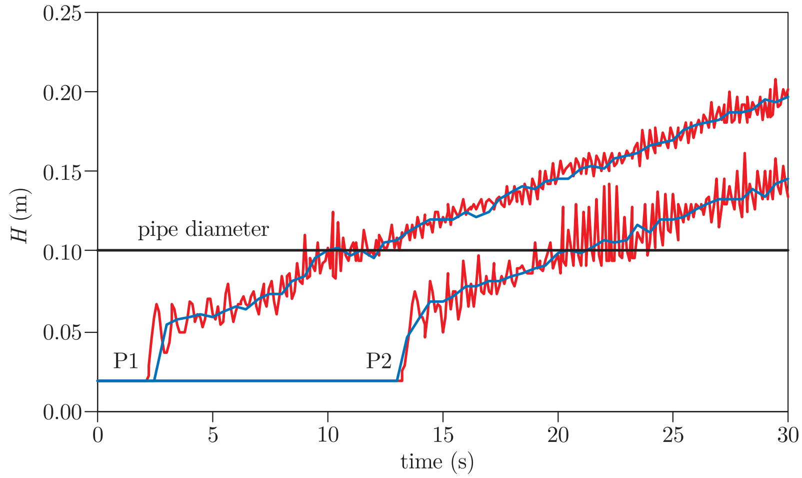

Figure 14. Test No. 3: Piezometric head calculated using classic (red line) and improved (blue line) McCormack scheme

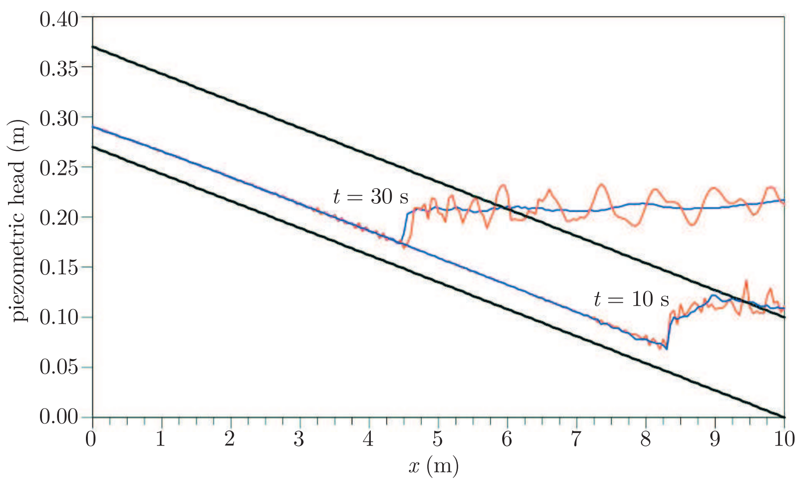

Figure 15. Test No. 3: Piezometric profiles calculated using classic (red line) and improved (blue line) McCormack scheme

pressurized flow behind. Once the surge reached the downstream end, there was full flow in the entire pipe. Pressure measurements were taken at selected locations. In this test the measurements from point P5, located 2.5 m from the upstream end, were used for comparison with the calculations.

The pipe was discretized into 201 nodes with the spatial step length of $\Delta x = 0.03$ m to simulate the flow. The simulation was carried out with the time step of $\Delta t = 0.001$ s, ensuring the stability of the solution. The comparison between measurements and calculations published by Fuamba [28] is presented in Figure 16. The calculations presented in this graph were obtained using the method of characteristics. The results of the authors' own numerical simulation are shown in Figure 17. The agreement between the results is not very satisfactory but the main features of the transient flow were predicted reasonably well. The pressurization moment and piezometric head at control point P5 were simulated precisely enough, however, the shape of pressure variation in time differed. This disagreement cannot be analyzed in detail at this moment due to the lack of certain information about the experiment conditions.

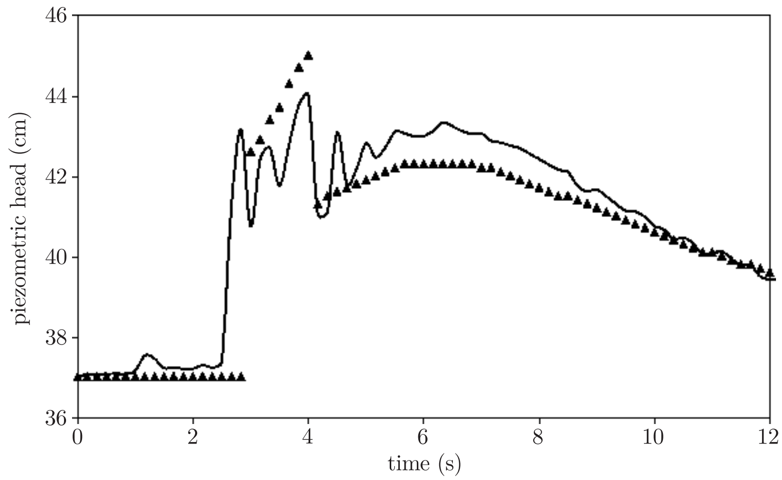

Figure 16. Test No. 4: Piezometric head above local reference level measured (line) and calculated (markers) by Fuamba [28] at control point P5

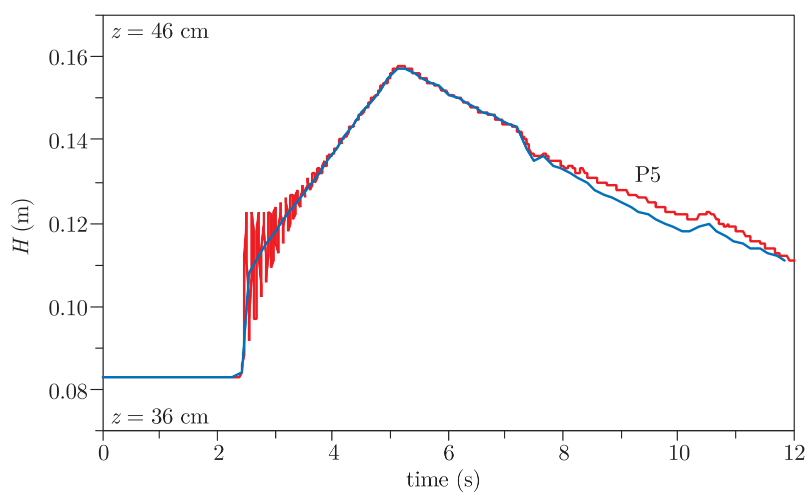

Figure 17. Test No. 4: Piezometric head above pipe bottom at P5 calculated using classic (red line) and improved (blue line) McCormack scheme

Additionally, the evolution of the pipe pressurization in time is presented in Figure 18. It can be found that this process was very fast and the results of simulations predicted the experiment properly. This last test case proved that the

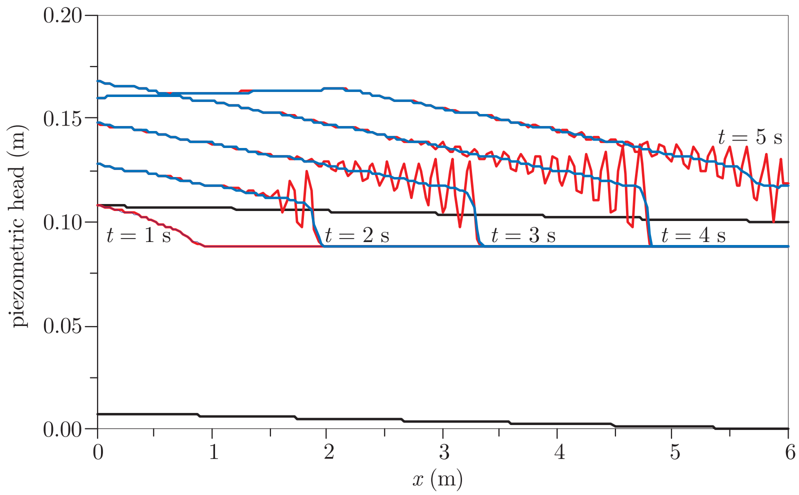

Figure 18. Test No. 4: Piezometric profiles calculated using classic (red line) and improved (blue line) McCormack scheme

improved McCormack scheme was devoid of unphysical oscillations and ensured better simulation results for transient and transcritical flow in pipes than the standard method.

5. Conclusions

The main aim of this research was to provide a robust model for simulating transient and transcritical flow in storm sewer systems using the 'Preissmann slot' approach for the treatment of pressurized flows. The numerical solution of the Saint–Venant equations for one-dimensional flow based on the standard and improved McCormack scheme was investigated in the paper. The original scheme improvement was based on the theory of total variation diminishing schemes. The results of numerical simulations of the flow in a single pipe of a storm sewer were shown and analyzed. They were compared with the laboratory measurements published in the technical papers. The following conclusions can be drawn out from the research:

- The spurious oscillations of the calculated results obtained using the standard McCormack scheme make the solution unphysical. The improved scheme is capable of capturing sharp fronts without generating oscillations. The modification is easily introduced into the standard McCormack scheme algorithm.
- The improved McCormack scheme together with idea of the 'Preissmann slot' allows modeling the transient and transcritical flow in pipes of storm sewers. It quite accurately describes the main flow features, such as positive and negative open channel pressurized flow interfaces. The improved method better predicts the flow parameters than the standard algorithm.

- Results from several numerical simulations compared with the published measurements show that the overall performance of the method can be considered as a very good candidate for modeling the flow in storm sewer systems. It seems that the improved McCormack scheme can be incorporated into the integrated model of urban flooding.
- The test cases presented in this paper allow the numerical simulation results to be seen as satisfying. However, as no detailed information about the conditions of experiments and no exact laboratory data are available for the authors of this paper, the model cannot be treated as verified. One's own measurements should be used for comparison in order to asses the numerical solution better than on qualitative level only. The laboratory experiments will be carried out in the second part of the research project at the hydraulic laboratory of the Gdansk University of Technology.

Acknowledgements

The authors wish to acknowledge the financial support offered by the Polish National Science Center for the research project N N523 745840 and the Regional Fund for Environmental Protection and Water Management in Gdansk (WFO/SGW) for the project RX-03/12/2011.

References

[1] Cunge J A, Holly Jr F M and Verwey A 1980 *Practical Aspects of Computational River Hydraulics*, Pitman, London
[2] Wiggert D C 1972 *J. Hydraul. Division* **98** (1) 11
[3] Song C C S, Cardle J A and Leung K S 1983 *J. Hydraul. Eng.* **109** (11) 1487
[4] Vasconcelos J G, Wright S J and Roe P L 2006 *J. Hydraul. Eng.* **132** (6) 553
[5] Li J and McCorquodale A 1999 *J. Hydraul. Eng.* **125** (11) 1170
[6] Cardle J A, Song C C S and Yuan M 1989 *J. Hydraul. Eng.* **115** (2) 169
[7] Politano M, Odgaard A J and Klecan W 2007 *J. Hydraul. Eng.* **133** (10) 1103
[8] Garcia-Navarro P, Priestley A and Alcrudo F 1994 *J. Hydraul. Res.* **32** (5) 721
[9] Capart H, Sillen X and Zech Y 1997 *J. Hydraul. Res.* **35** (5) 659
[10] Trajkovic B, Ivetic M, Calomino F and D'Ippolito A 1999 *Wat. Sci. Tech.* **39** (9) 105
[11] Abbott M B and Basco D R 1989 *Computational Fluid Dynamics*, Longman Scientific & Technical, Essex
[12] Szymkiewicz R 2010 *Numerical Modeling in Open Channel Hydraulics*, Springer, London
[13] Bourdarias C and Gerbi S 2007 *J. Comput. Appl. Math.* **209** (1) 109
[14] Sanders B F and Bradford S F 2011 *J. Hydraul. Eng.* **137** (2) 158
[15] Wright S J, Creech C T, Lewis J M. and Vasconcelos J G 2008 *Environ. Fluid Mech.* **8** 605
[16] Toro E F 1997 *Riemann Solvers and Numerical Methods for Fluid Dynamics*, Springer-Verlag Berlin
[17] Leon A S, Ghidaoui M S, Schmidt A R and Garcia M H 2009 *J. Hydraul. Res.* **47** (2) 147
[18] Garcia-Navarro P, Alcrudo F, Saviron J M 1992 *J. Hydraul. Eng.* **118** 1359
[19] Tseng M H and Chu C R 2000 *Adv. Wat. Res.* **23** 637
[20] Papanicolaou A N, Bdour A and Wicklein E 2004 *J. Hydraul. Res.* **42** (2) 357
[21] Ji Z 1998 *J. Hydraul. Eng.* **124** (3) 307
[22] Potter D 1982 *Computational Physics*, PWN, Warsaw
[23] Harten A and Hyman J M 1983 *J. Comput. Phys.* **50** 235

[24] Tseng M H 2003 *Int. J. Numer. Meth. Fluids* **43** 71
[25] Delis A I and Skeels C P 1998 *Int. J. Numer. Meth. Fluids* **26** 791
[26] Coulson C A and Jeffrey A 1982 *Waves – Mathematical Models*, WNT, Warsaw
[27] Roe P L 1981 *J. Comput. Phys.* **43** 357
[28] Fuamba M 2002 *J. Hydraul. Res.* **40** (6) 685